Introduction to Finite Element

ENGR 45901/55901/75901

Spring Semester

Dr. A. Aziz

College of Engineering and Aeronautics

KENT STATE UNIVERSITY

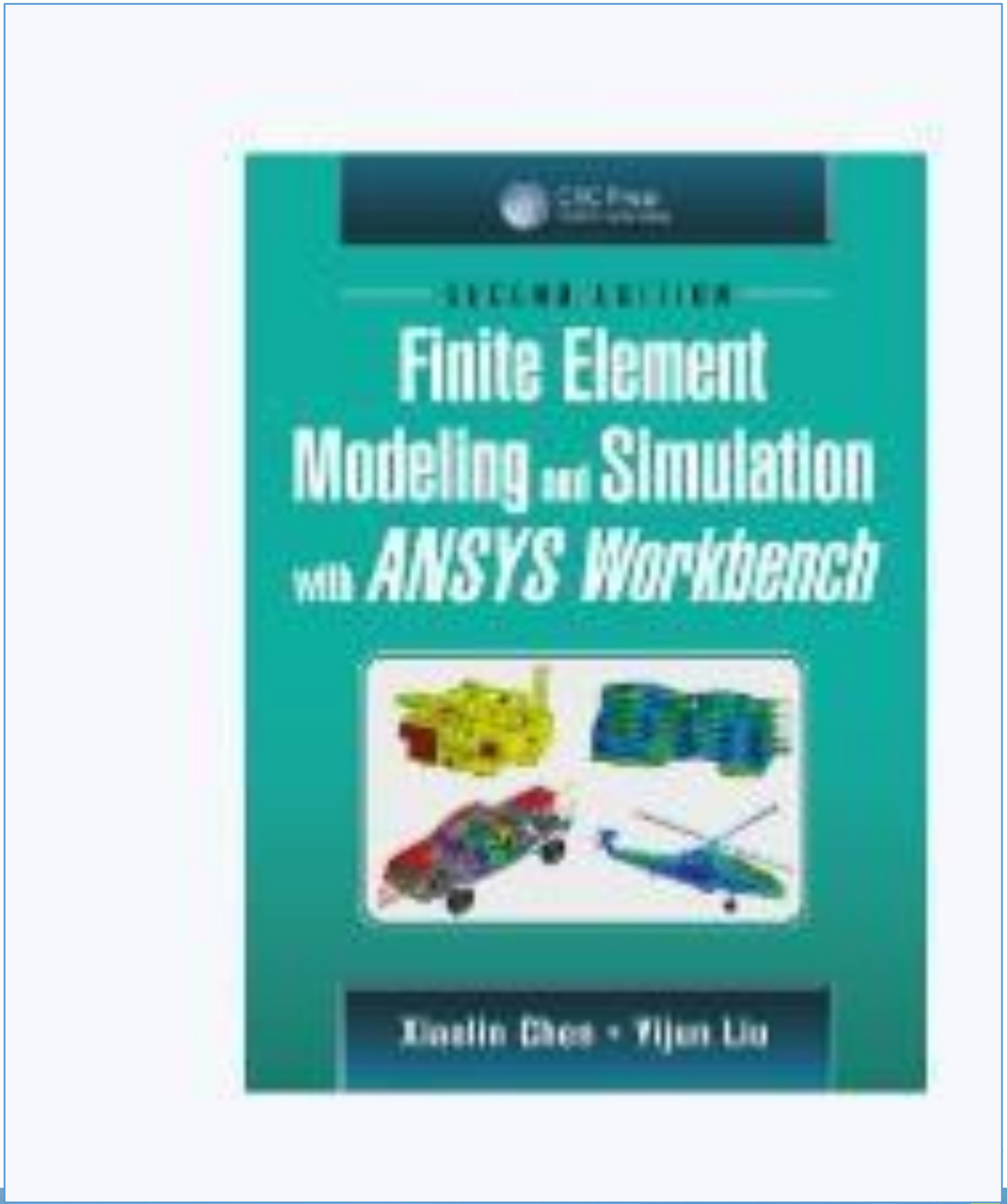
Chapter 2
Bars and Trusses

Introduction to Finite Element

ENGR 45901/55901/75901
Spring Semester

Dr. A. Aziz

College of Engineering and Aeronautics



2.1 Introduction

This chapter introduces you to the simplest one-dimensional (1-D) structural element, namely the bar element, and the FEA of truss structures using such element.

- ❑ Trusses are commonly used in the design of buildings, bridges, and towers (Figure 2.1). They are triangulated frameworks composed of slender bars whose ends are connected through bolts, pins, rivets, and so on.
- ❑ Truss structures create large, open, and uninterrupted space, and offer lightweight and economical solutions to many engineering situations.
- ❑ If a truss, along with the applied load, lies in a single plane, it is called a planar truss.
- ❑ it has members and joints extending into the three-dimensional (3-D) space, it is then a space truss.

FIGURE 2.1

Truss examples: (a) Montreal Biosphere Museum (http://en.wikipedia.org/wiki/Montreal_Biosph%C3%A8re). (b) Betsy Ross Bridge (http://en.wikipedia.org/wiki/Betsy_Ross_Bridge).

Most structural analysis problems such as stress and strain analysis can be treated as linear static problems, based on the following assumptions:

1. Small deformations (loading pattern is not changed due to the deformed shape)
2. Elastic materials (no plasticity or failures)
3. Static loads (the load is applied to the structure in a slow or steady fashion)
4. Linear analysis can provide most of the information about the behavior of a structure and can be a good approximation for many analyses. It is also the basis of nonlinear FEA in most of the cases.
5. In Chapters 2 through 7, only linear static responses of structures are considered.

KENT STATE UNIVERSITY

2.2 Review of the 1-D Elasticity Theory

We begin by examining the problem of an axially loaded bar based on 1-D linear elasticity.

Consider a uniform prismatic bar shown in Figure 2.2. The parameters L, A, and E are the length, cross-sectional area, and elastic modulus of the bar, respectively.

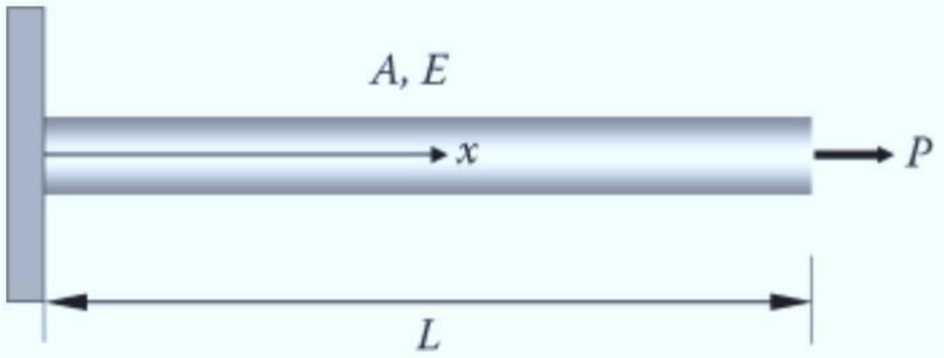


FIGURE 2.2
An axially loaded elastic bar.

Let u, ε, and σ be the displacement, strain, and stress, respectively (all in the axial direction and functions of x only), we have the following basic relations:

Strain–displacement relation:

$$\boxed{\varepsilon(x) = \frac{du(x)}{dx}} \qquad (2.1)$$

Stress–strain relation:

$$\boxed{\sigma(x) = E\varepsilon(x)} \qquad (2.2)$$

Equilibrium equation:

$$\boxed{\frac{d\sigma(x)}{dx} + f(x) = 0} \tag{2.3}$$

where $f(x)$ is the body force (force per unit volume, such as gravitational and magnetic forces) inside the bar. To obtain the displacement, strain, and stress field in a bar, Equations 2.1 through 2.3 need to be solved under given boundary conditions, which can be done readily for a single bar, but can be tedious for a network of bars or a truss structure made of many bars.

2.3 Modeling of Trusses

For the truss analysis, it is often assumed that:

(1) the bar members are of uniform cross sections and are joined together by frictionless pins, and

(2) loads are applied to joints only and not in between joints along the truss members.

(3) It is based on these assumptions that the truss members are considered to carry only axial loads and have negligible bending resistance.

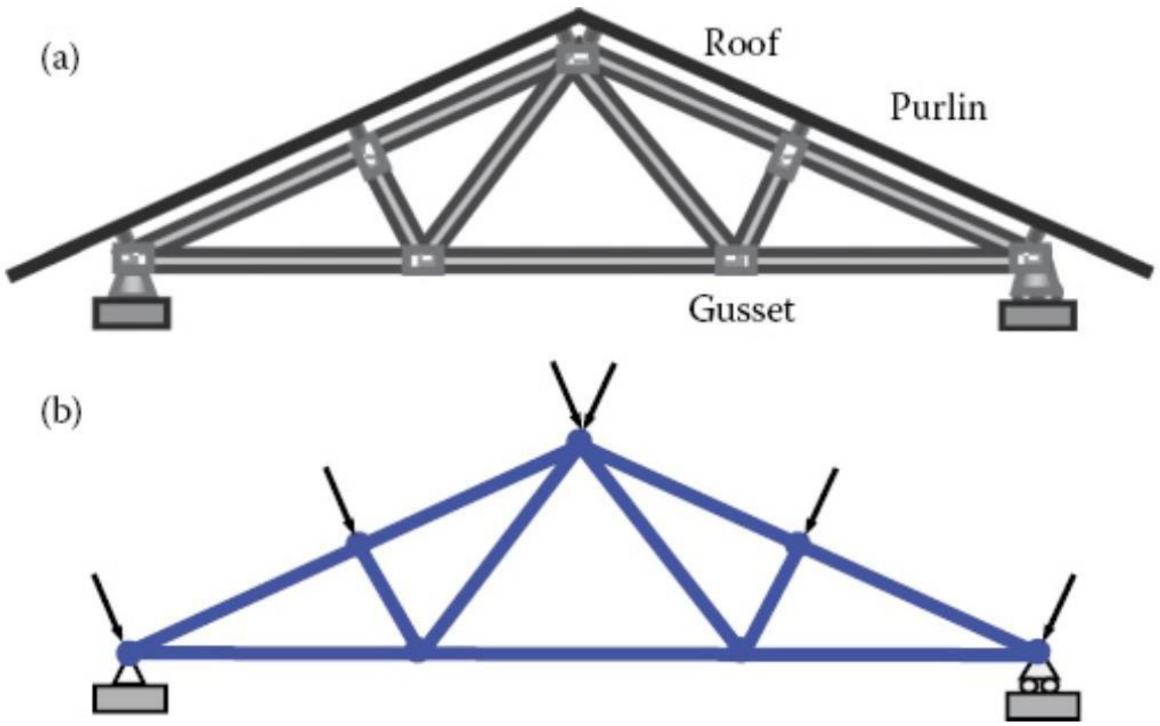


FIGURE 2.3
Modeling of a planar roof truss: (a) Physical structure. (b) Discrete model.

Figure 2.3Modeling of a planar roof truss: (a) Physical structure. (b) Discrete model.

Review of the 1-D Elasticity Theory

- Consider a uniform prismatic bar shown below

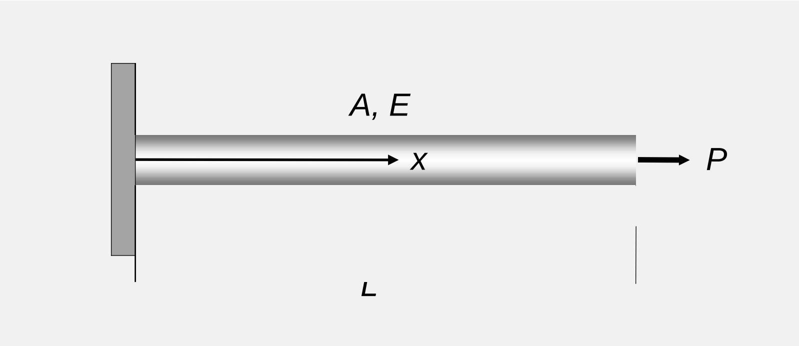


Figure 2.2. An axially loaded elastic bar.

Strain-displacement relation

$$\varepsilon(x) = \frac{du(x)}{dx}$$

Strain-displacement relation

$$\sigma(x) = E\,\varepsilon(x)$$

Equilibrium equation

$$\frac{d\sigma(x)}{dx} + f(x) = 0$$

The displacement, strain and stress field in a bar needs to be solved under given boundary conditions, which can be done readily for a single bar, but can be tedious for a network of bars or a truss structure made of many bars.

A truss is an assembly of axial bars …

Truss Modeling & Bar Element Formulation

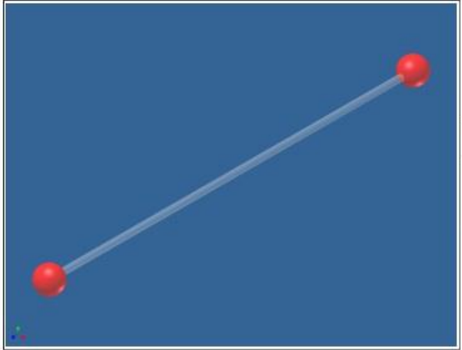
FEA Geometry Element Types

- Elements fall into four major categories: 2D line elements, 2D planar elements, and 3D solid elements which are all used to define geometry; and special elements used to apply boundary conditions.

- For example, special elements might include gap elements to specify a gap between two pieces of geometry. Spring elements are used to apply a specific spring constant at a specified node or set of nodes.

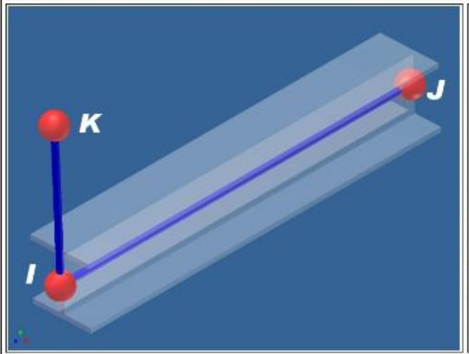
- Rigid elements are used to define a rigid connection to or within a model. The figures below show nodes in red and the element in translucent blue except for the beam element which is bright blue. The most common geometry elements are show below.

- Most FEA tools support additional element types as well as somewhat different implementations of even these common elements.



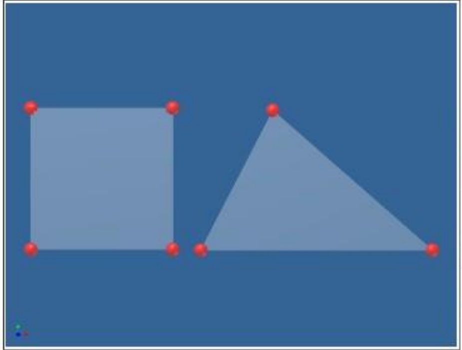
Truss Element (2D Line)

Truss elements are long and slender, have 2 nodes, and can be oriented anywhere in 3D space. Truss elements transmit force axially only and are 3 DOF elements which allow translation only and not rotation. Trusses are normally used to model towers, bridges, and buildings. A constant cross section area is assumed and they are used for linear elastic structural analysis.



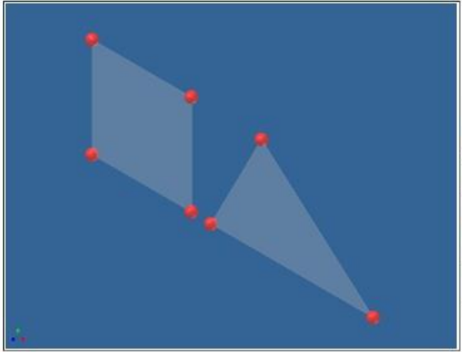
Beam Element (2D Line)

Beam elements are long and slender, have three nodes, and can be oriented anywhere in 3D space. Beam elements are 6 DOF elements allowing both translation and rotation at each end node. That is the primary difference between beam and truss elements. The I J nodes define element geometry, the K node defines the cross sectional orientation. This is how you differentiate between the strong and weak axis of bending for a beam. A constant cross section area is assumed. In the image, the beam shape is shown only for visualization, the element is the dark blue rod. The I J axis runs from the near to far node. K is shown vertically above the I node or could be horizontally to the right of I.



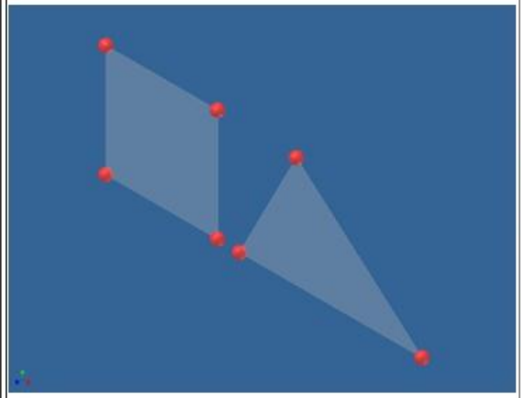
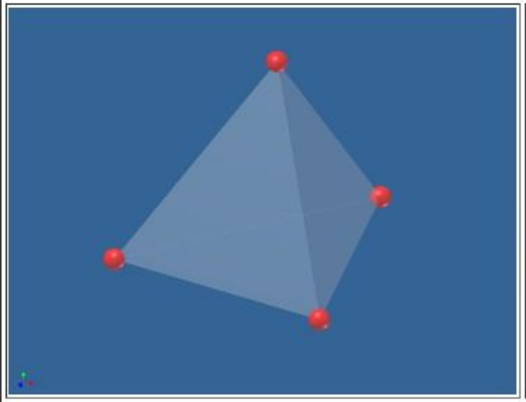
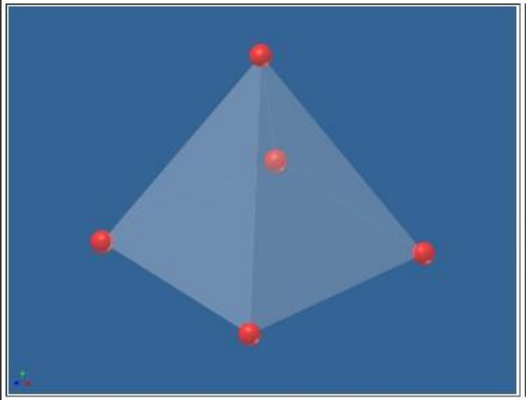
2D Element (2D Planar)

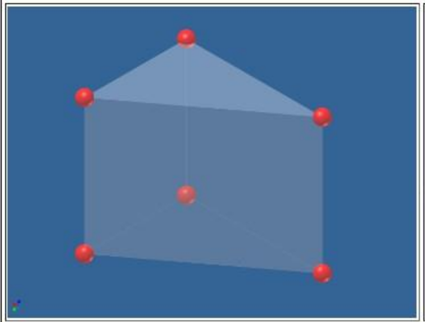
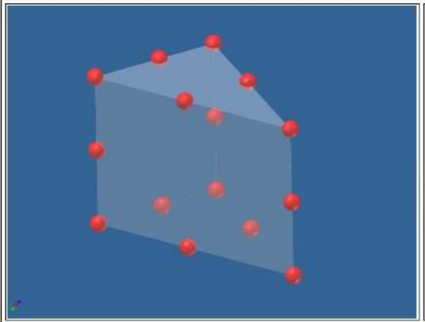
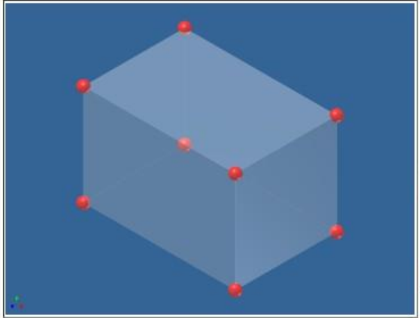
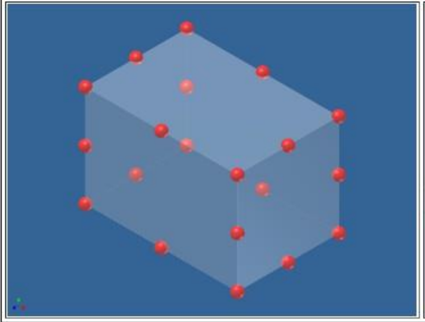
2D Elements are 3 or 4 node elements with only 2 DOF, Y and Z translation, and are normally created in the YZ plane. They are used for Plane Stress or Plane Strain analyses. Common applications include axisymmetric bodies of revolution such as missile radomes, radial seals, etc. and long sections with constant cross sectional area such as a dam. Plane Stress implies no stress normal to the cross section defined - strain is allowed - suitable to model the 2D cross section of a body of revolution. Plane Strain implies no strain normal to the cross section defined - stress is allowed - suitable to model the 2D cross section of a long dam.



Membrane Element (2D Planar)

Membrane Elements are 3 or 4 node 2D elements that can be oriented anywhere in 3D space. They can be used to model thin membrane like materials like fabric, thin metal shells, etc. These elements will not support or transmit a moment load or stress normal to the surface. They support only translational DOF not rotational and in-plane loading. The thickness of the membrane must be small relative to its length or width. Membrane thickness is defined as a fixed parameter which can be varied. The geometry is drawn at the midplane with zero thickness shown, similar to a plate element.

	**Plate / Shell Element (2D Planar)** Plate / Shell elements are 3 or 4 node 2D planar elements that can be oriented anywhere in 3D space. They are typically used to model structures comprised of shells such as pressure vessels, automobile bodies, ship hulls, and aircraft fuselages. Generally a thicker wall than for a membrane element but about 1/10 the length or width. All translational DOF are supported as well as rotational DOF that are not out of plane. That is rotation about the normal to the element surface is not allowed. Plate thickness is defined as a fixed parameter which can be varied. The geometry is drawn at the midplane with zero thickness shown. Different FEA tools call these either Plates or Shells. This is the preferred element type for most thin walled structures.
	**3D Tetrahedra Element, 4 Nodes (3D Solid)** See definition below for the 8 node brick, you can usually specify either all tetrahedra, all bricks, or a mixture of both with some automatic mesh generators. A mix of tets and bricks usually produces a higher fidelity model.
	**3D Tetrahedra Element, 5 Nodes, Pyramid (3D Solid)** See definition below for the 8 node brick, you can usually specify either all tetrahedra, all bricks, or a mixture of both with some automatic mesh generators.

	**3D Tetrahedra Element, 6 Nodes, Wedge (3D Solid)** See definition below for the 8 node brick, you can usually specify either all tetrahedra, all bricks, or a mixture of both with some automatic mesh generators.
	**3D Tetrahedra Element with Midside Nodes, 15 Nodes, Wedge (3D Solid)** See definition below for the 8 node brick, you can usually specify either all tetrahedra, all bricks, or a mixture of both with most automatic mesh generators.
	**3D Brick Element, 8 Nodes (3D Solid)** Brick or tetrahedra elements may have 4, 5, 6, 7, 8, 15, or 20 nodes and support only translational DOF. They are normally used to model solid objects for which plate elements are not appropriate. You can usually specify either all tetrahedra, all bricks, or a mixture of both with some automatic mesh generators. This is the most common, and frequently the only element type supported by automatic mesh generators. Bricks work quite well for any "blocky" structures which are typical of machined, cast, or forged fabricated parts. Structural and thermal bricks exist so the same model geometry can be used for both the initial steady state heat transfer and subsequent thermal stress computations. Bricks compute stress through the thickness of a part.
	**3D Brick Element with Midside Nodes, 20 Nodes (3D Solid)** See definition above for the 8 node brick. Midside nodes can be included if desired, also some FEA tools include an additional 21st node at the centroid of the brick which can be useful in computation quality comparisons.

Modeling of Trusses

- For the truss analysis, it is often assumed that
  - The bars are of uniform cross sections and joined by frictionless pins.
  - Loads are applied to joints only.
- Based on the assumptions, truss members are considered to carry only axial loads and have negligible bending resistance.

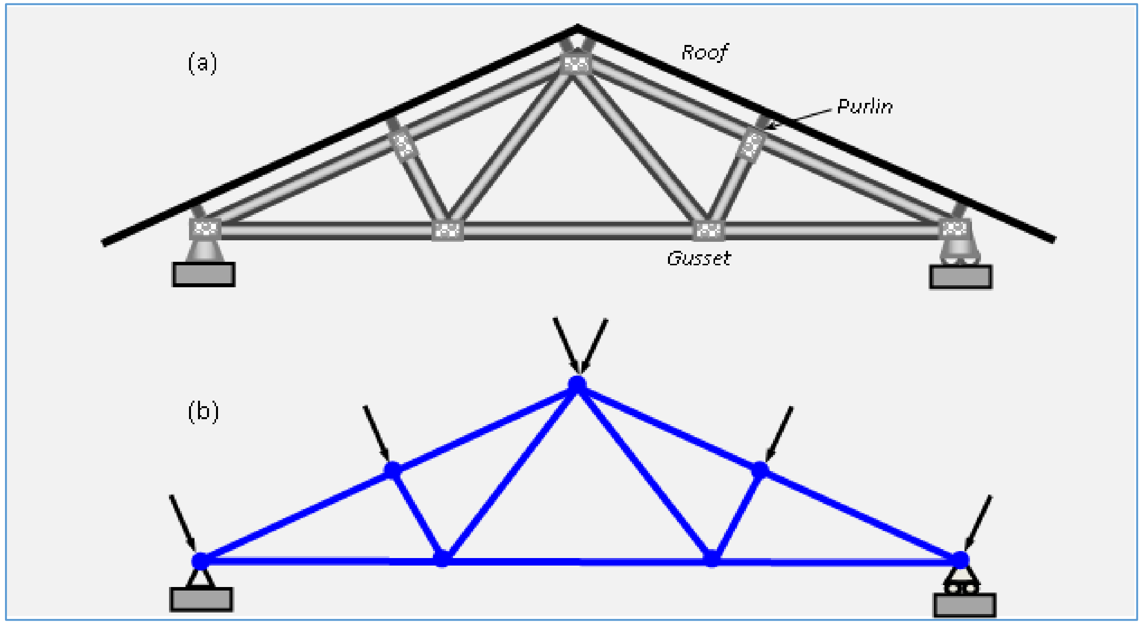


Figure 2.3. Modeling of a planar roof truss: (a) Physical structure (b) Discrete model.

https://www.youtube.com/watch?v=sKZAipnrV9E

Truss Analysis Ansys Workbench

Finite Element Modeling and Simulation with ANSYS Workbench

Formulation of the Bar Element

❑ Stiffness Matrix – Direct Method

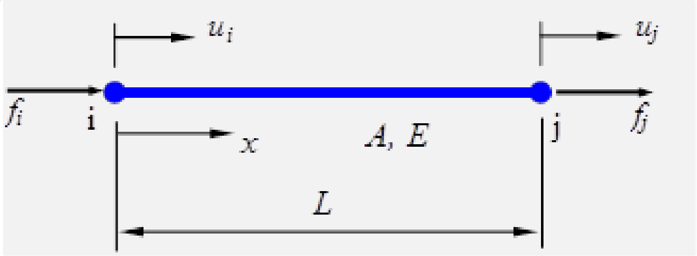


Figure 2.4. Notation for a bar element.

We have $$u(x) = \left(1 - \frac{x}{L}\right)u_i + \frac{x}{L}u_j$$

$$\varepsilon = \frac{u_j - u_i}{L} = \frac{\Delta}{L}$$

$$\sigma = E\varepsilon = \frac{E\Delta}{L} \quad \text{and} \quad \sigma = \frac{F}{A}$$

Therefore $$F = \frac{EA}{L}\Delta = k\Delta$$

We conclude that the bar behaves like a spring. The element stiffness matrix is:

$$\mathbf{k} = \begin{bmatrix} k & -k \\ -k & k \end{bmatrix} = \begin{bmatrix} \frac{EA}{L} & -\frac{EA}{L} \\ -\frac{EA}{L} & \frac{EA}{L} \end{bmatrix}$$

Element equilibrium equation is:

$$\boxed{\frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = \begin{Bmatrix} f_i \\ f_j \end{Bmatrix}}$$

Degree of Freedom (DOF): Number of components of the displacement vector at a node. For 1-D bar element along the *x*-axis, we have one DOF at each node.

Formulation of the Bar Element

❑ Stiffness Matrix – Energy Approach

Define two *linear shape functions* as follows

$$N_i(\xi) = 1 - \xi, \qquad N_j(\xi) = \xi \quad \text{where} \quad \xi = \frac{x}{L}, \qquad 0 \le \xi \le 1$$

We can write

$$u = \begin{bmatrix} N_i & N_j \end{bmatrix} \begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = \mathbf{Nu} \quad \text{and} \quad \varepsilon = \frac{du}{dx} = \left[\frac{d}{dx} \mathbf{N} \right] \mathbf{u} = \mathbf{Bu}$$

where **B** is the element *strain-displacement matrix* $\mathbf{B} = [-1/L \quad 1/L]$

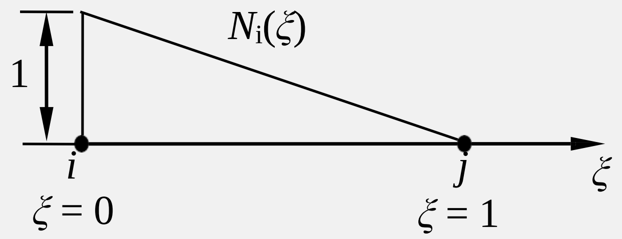


Figure 2.5. The shape functions for a bar element.

What are element shapes in FEA?

- ❑ Elements in FEA are generally grouped into **1D element, 2D element, and 3D element**. They are recognized based on their shapes.
- ❑ For example, elements can take on the form of a straight line or curve, triangle or quadrilateral, tetrahedral and many more. The simplest element is a line made of two nodes.

Formulation of the Bar Element

Stress can be written as

$$\sigma = E\varepsilon = E\mathbf{B}\mathbf{u}$$

Consider the stored *strain energy*

$$U = \frac{1}{2}\int_V \sigma^T \varepsilon dV = \frac{1}{2}\int_V \left(\mathbf{u}^T\mathbf{B}^T E\mathbf{B}\mathbf{u}\right)dV$$

$$= \frac{1}{2}\mathbf{u}^T\left[\int_V \left(\mathbf{B}^T E\mathbf{B}\right)dV\right]\mathbf{u}$$

The *potential* of the external forces is

$$\Omega = -f_i u_i - f_j u_j = -\mathbf{u}_T\mathbf{f}$$

The total potential of the system is

$$\Pi = U + \Omega$$

$$\Pi = \frac{1}{2}\mathbf{u}^T\left[\int_V \left(\mathbf{B}^T E\mathbf{B}\right)dV\right]\mathbf{u} - \mathbf{u}^T\mathbf{f}$$

Setting $d\Pi = 0$ by the principle of minimum potential energy, we obtain

$$\mathbf{k}\mathbf{u} = \mathbf{f}$$

where $\mathbf{k} = \int_V \left(\mathbf{B}^T E\mathbf{B}\right)dV$

For the bar element

$$\mathbf{k} = \int_0^L \begin{Bmatrix} -1/L \\ 1/L \end{Bmatrix} E\begin{bmatrix} -1/L & 1/L \end{bmatrix} A dx = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

Formulation of the Bar Element

❑ Treatment of Distributed Load

Distributed axial load *q* can be converted to two equivalent nodal forces using the shape functions. Consider the work done by the distributed load *q.*

$$W_q = \frac{1}{2}\int_0^L u(x)q(x)dx = \frac{1}{2}\int_0^L (\mathbf{Nu})^T q(x)dx = \frac{1}{2}\begin{bmatrix} u_i & u_j \end{bmatrix}\int_0^L \begin{bmatrix} N_i(x) \\ N_j(x) \end{bmatrix} q(x)dx$$

$$= \frac{1}{2}\mathbf{u}^T \int_0^L \mathbf{N}^T q(x)dx$$

The work done by the equivalent nodal forces are

$$W_{f_q} = \frac{1}{2} f_i^q u_i + \frac{1}{2} f_j^q u_j = \frac{1}{2}\mathbf{u}^T \mathbf{f}_q$$

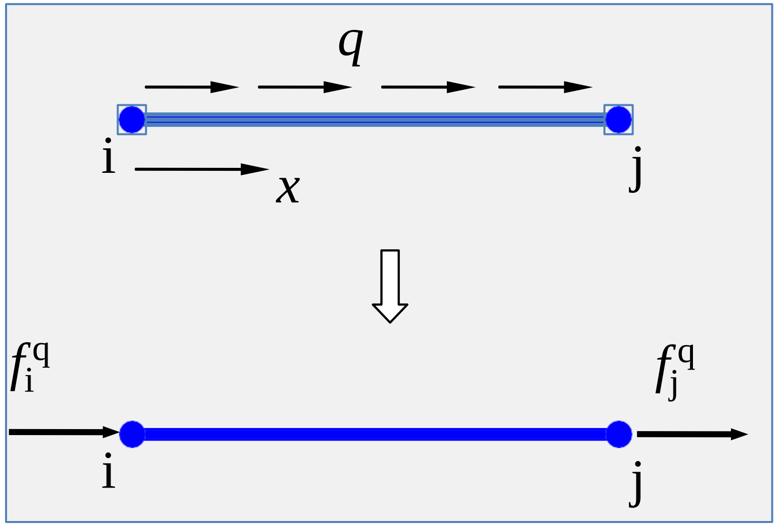


Figure 2.6. Conversion of a distributed load on one element.

Formulation of the Bar Element

Setting $W_q = W_{f_q}$, we obtain the equivalent nodal force vector

$$\mathbf{f}_q = \begin{Bmatrix} f_i^q \\ f_j^q \end{Bmatrix} = \int_0^L \mathbf{N}^T q(x)dx = \int_0^L \begin{bmatrix} N_i(x) \\ N_j(x) \end{bmatrix} q(x)dx$$

If *q* is a *constant*, we have

$$\mathbf{f}_q = q\int_0^L \begin{bmatrix} 1 - x/L \\ x/L \end{bmatrix} dx = \begin{Bmatrix} qL/2 \\ qL/2 \end{Bmatrix}$$

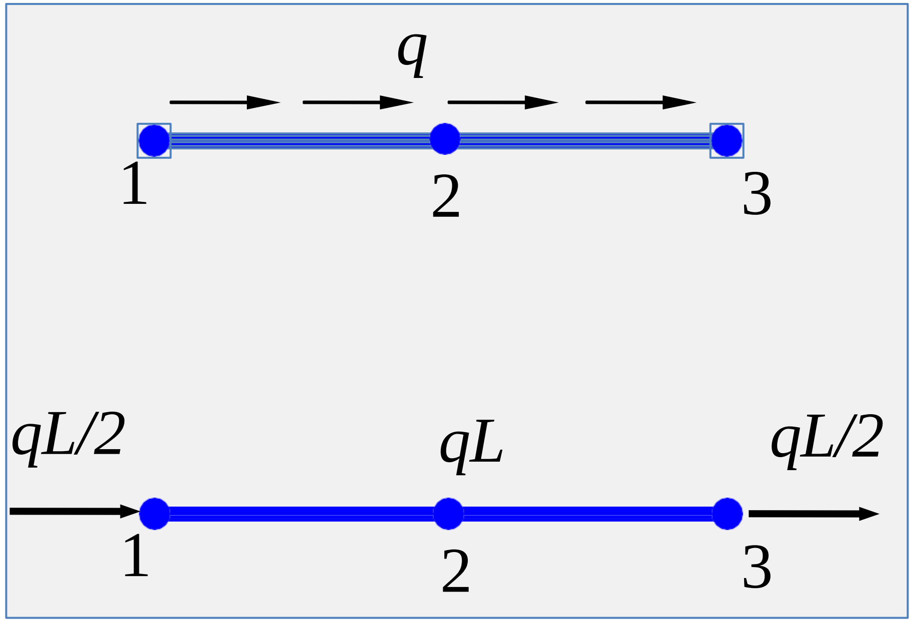


Figure 2.7. Conversion of a distributed load with constant intensity *q* on two elements.

Formulation of the Bar Element

- Bar Element In 2-D and 3-D

2-D Case

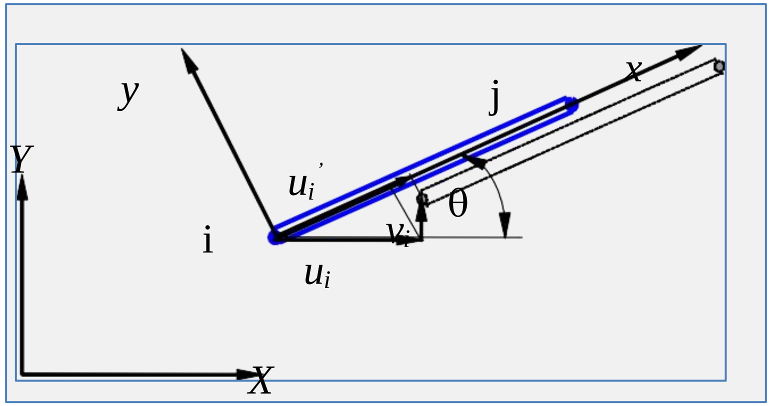


Figure 2.8. Local and global coordinates for a bar in 2-D space.

Local	*Global*
x, y	X, Y
u_i', v_i'	u_i, v_i
1 DOF at each node	2 DOFs at each node

Formulation of the Bar Element

Displacement vectors in the local and global coordinates are related as follows

$$u_i' = u_i \cos\theta + v_i \sin\theta = \begin{bmatrix} l & m \end{bmatrix} \begin{Bmatrix} u_i \\ v_i \end{Bmatrix}$$

$$v_i' = -u_i \sin\theta + v_i \cos\theta = \begin{bmatrix} -m & l \end{bmatrix} \begin{Bmatrix} u_i \\ v_i \end{Bmatrix}$$

where $l = \cos\theta, m = \sin\theta$

In matrix form

$$\begin{Bmatrix} u_i' \\ v_i' \end{Bmatrix} = \begin{bmatrix} l & m \\ -m & l \end{bmatrix} \begin{Bmatrix} u_i \\ v_i \end{Bmatrix}$$ or $$\mathbf{u}_i' = \tilde{\mathbf{T}}\mathbf{u}_i$$

For the two nodes of the bar element, we have

$$\begin{Bmatrix} u_i' \\ v_i' \\ u_j' \\ v_j' \end{Bmatrix} = \begin{bmatrix} l & m & 0 & 0 \\ -m & l & 0 & 0 \\ 0 & 0 & l & m \\ 0 & 0 & -m & l \end{bmatrix} \begin{Bmatrix} u_i \\ v_i \\ u_j \\ v_j \end{Bmatrix}$$

In matrix form

$$\mathbf{u}' = \mathbf{T}\mathbf{u}$$ with $$\mathbf{T} = \begin{bmatrix} \tilde{\mathbf{T}} & \mathbf{0} \\ \mathbf{0} & \tilde{\mathbf{T}} \end{bmatrix}$$

Formulation of the Bar Element

In the local coordinate system, we have

$$\frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u'_i \\ u'_j \end{Bmatrix} = \begin{Bmatrix} f'_i \\ f'_j \end{Bmatrix}$$

Augmenting this equation, we write

$$\frac{EA}{L}\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}\begin{Bmatrix} u'_i \\ v'_i \\ u'_j \\ v'_j \end{Bmatrix} = \begin{Bmatrix} f'_i \\ 0 \\ f'_j \\ 0 \end{Bmatrix}$$

or $$\mathbf{k}'\mathbf{u}' = \mathbf{f}'$$

Formulation of the Bar Element

Using the transformations, we obtain

$$\mathbf{k}'\mathbf{T}\mathbf{u} = \mathbf{T}\mathbf{f}$$

$$\mathbf{T}^T\mathbf{k}'\mathbf{T}\mathbf{u} = \mathbf{f}$$

Thus, the element stiffness matrix **k** in the global coordinate system is

$$\mathbf{k} = \mathbf{T}^T\mathbf{k}'\mathbf{T}$$

The explicit form

$$\mathbf{k} = \frac{EA}{L}\begin{array}{c} \begin{array}{cccc} u_i & v_i & u_j & v_j \end{array} \\ \begin{bmatrix} l^2 & lm & -l^2 & -lm \\ lm & m^2 & -lm & -m^2 \\ -l^2 & -lm & l^2 & lm \\ -lm & -m^2 & lm & m^2 \end{bmatrix} \end{array}$$

Formulation of the Bar Element

3-D Case

A degree of freedom corresponds to a translation or a rotation at each node of an element. There can be up to 6 degrees of freedom per node depending on the element type. 1 direction translation (often X translation) 2 direction translation (often Y translation) 3 direction translation (often Z translation)

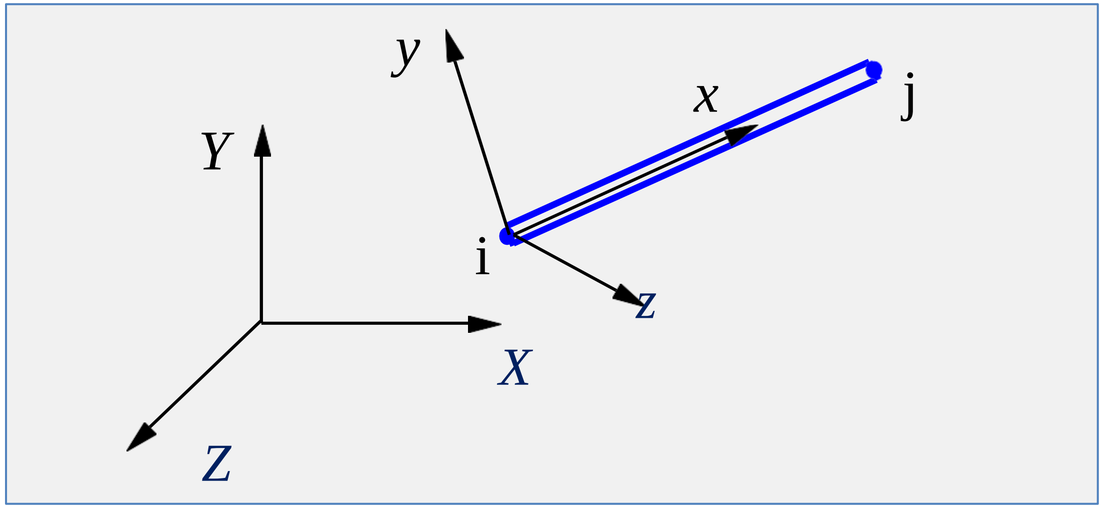


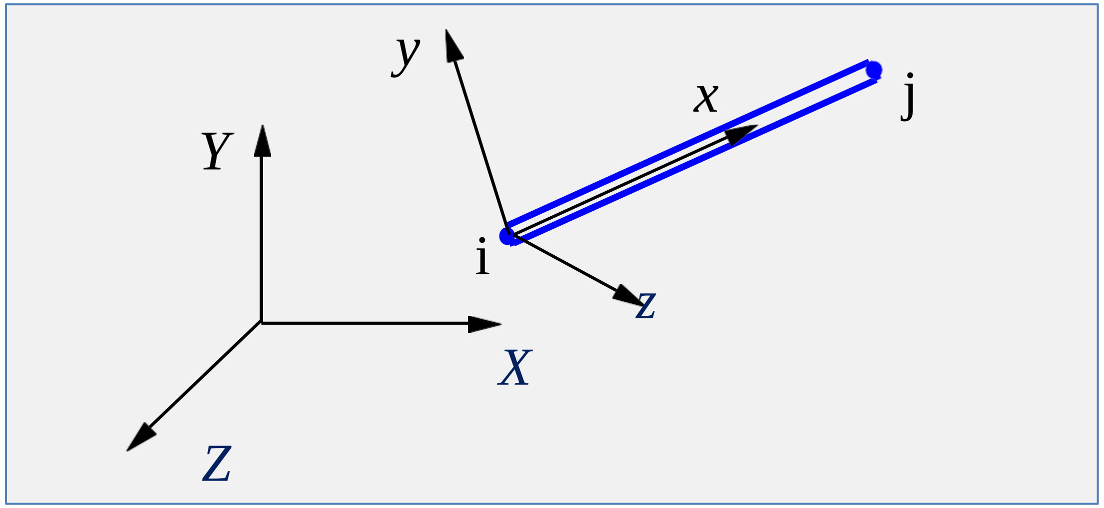
Figure 2.9. Local and global coordinates for a bar in 3-D space.

Local	*Global*
x, y, z	X, Y, Z
u'_i, v'_i, w'_i	u_i, v_i, w_i
1 DOF at each node	3 DOFs at each node

Formulation of the Bar Element

What is the degree of freedom in a finite element?

Degrees of freedom (DOF) are the most basic variables solved for in finite element analysis. Each element group has different degrees of freedom. A degree of freedom corresponds to a translation or a rotation at each node of an element.



ure 2.9. Local and global coordinates for a bar in 3-D space.

3-D Case

Local	*Global*
x, y, z	X, Y, Z
u'_i, v'_i, w'_i	u_i, v_i, w_i
1 DOF at each node	3 DOFs at each node

Formulation of the Bar Element

The transformation relation is

$$\begin{Bmatrix} u_i' \\ v_i' \\ w_i' \end{Bmatrix} = \begin{bmatrix} l_X & l_Y & l_Z \\ m_X & m_Y & m_Z \\ n_X & n_Y & n_Z \end{bmatrix} \begin{Bmatrix} u_i \\ v_i \\ w_i \end{Bmatrix}$$

where (l_X, l_Y, l_Z), (m_X, m_Y, m_Z) and (n_X, n_Y, n_Z) are the direction cosines of the local *x*, *y* and *z* coordinate axis in the global coordinate system, respectively.

FEM software packages will do this transformation automatically.

The input data for bar elements are simply the coordinates (*X, Y, Z*) for each node, *E* and *A* for each element (Length *L* can be computed from the coordinates of the two nodes).

Formulation of the Bar Element

- Element Stress

Once the nodal displacement is obtained for an element, the stress within the element can be calculated using the basic relations.

For 2-D cases, we proceed as follows

$$\sigma = E\varepsilon = E\mathbf{B}\begin{Bmatrix} u_i' \\ u_j' \end{Bmatrix} = E\begin{bmatrix} -\frac{1}{L} & \frac{1}{L} \end{bmatrix}\begin{bmatrix} l & m & 0 & 0 \\ 0 & 0 & l & m \end{bmatrix}\begin{Bmatrix} u_i \\ v_i \\ u_j \\ v_j \end{Bmatrix}$$

That is,

$$\sigma = \frac{E}{L}\begin{bmatrix} -l & -m & l & m \end{bmatrix}\begin{Bmatrix} u_i \\ v_i \\ u_j \\ v_j \end{Bmatrix}$$

which is a general formula for 2-D bar elements.

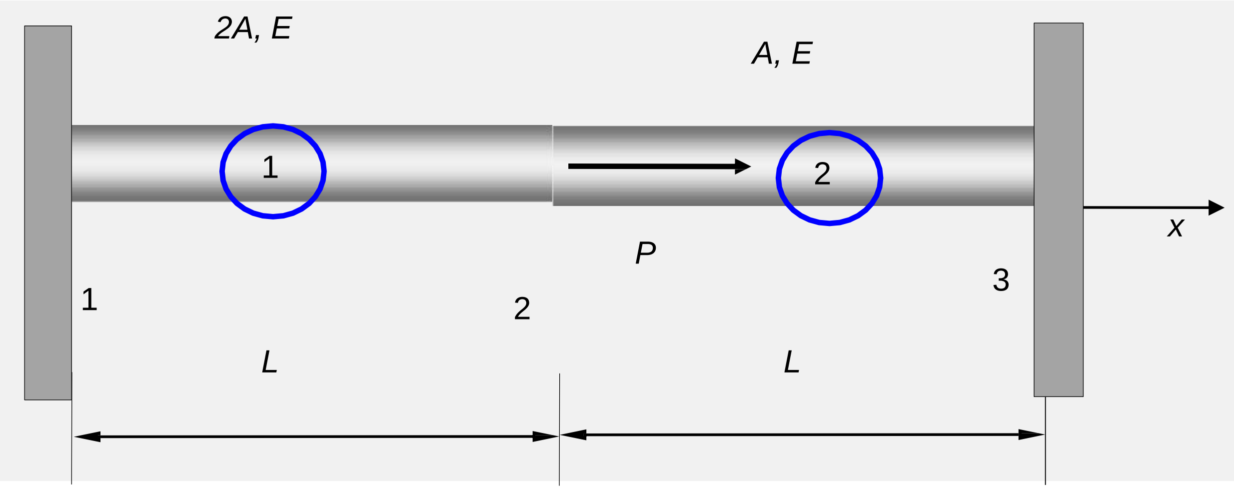
We now look at examples of 1-D stress and plane truss problems …

Examples with Bar Elements

Examples with Bar Elements

- Example Problems

Example 2.1



Problem: Find the stresses in the two-bar assembly which is loaded with force *P and* constrained at the two ends.

Solution: Use two 1-D bar elements.

Examples with Bar Elements

For element 1,

$$
\begin{array}{cc}
 & \begin{array}{cc} u_1 & u_2 \end{array} \\
\mathbf{k}_1 = \dfrac{2EA}{L} & \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}
\end{array}
$$

For element 2,

$$
\begin{array}{cc}
 & \begin{array}{cc} u_2 & u_3 \end{array} \\
\mathbf{k}_2 = \dfrac{EA}{L} & \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}
\end{array}
$$

Assemble the global FE equation as follows

$$
\frac{EA}{L}\begin{bmatrix} 2 & -2 & 0 \\ -2 & 3 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix}
$$

Examples with Bar Elements

Load and boundary conditions (BCs) are

$$u_1 = u_3 = 0, \qquad F_2 = P$$

FE equation becomes

$$\frac{EA}{L}\begin{bmatrix} 2 & -2 & 0 \\ -2 & 3 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} 0 \\ u_2 \\ 0 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ P \\ F_3 \end{Bmatrix}$$

Thus,

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \frac{PL}{3EA}\begin{Bmatrix} 0 \\ 1 \\ 0 \end{Bmatrix}$$

Examples with Bar Elements

Stress in element 1 is

$$\sigma_1 = E\varepsilon_1 = E\mathbf{B}_1\mathbf{u}_1 = E\begin{bmatrix}-1/L & 1/L\end{bmatrix}\begin{Bmatrix}u_1\\ u_2\end{Bmatrix}$$

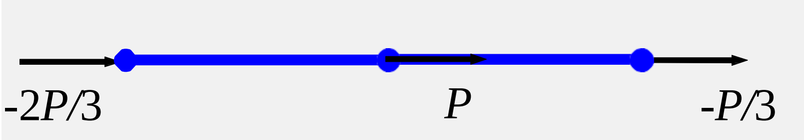
$$= E\frac{u_2 - u_1}{L} = \frac{E}{L}\left(\frac{PL}{3EA} - 0\right) = \frac{P}{3A}$$

Similarly, stress in element 2 is

$$\sigma_2 = E\varepsilon_2 = E\mathbf{B}_2\mathbf{u}_2 = E\begin{bmatrix}-1/L & 1/L\end{bmatrix}\begin{Bmatrix}u_2\\ u_3\end{Bmatrix}$$

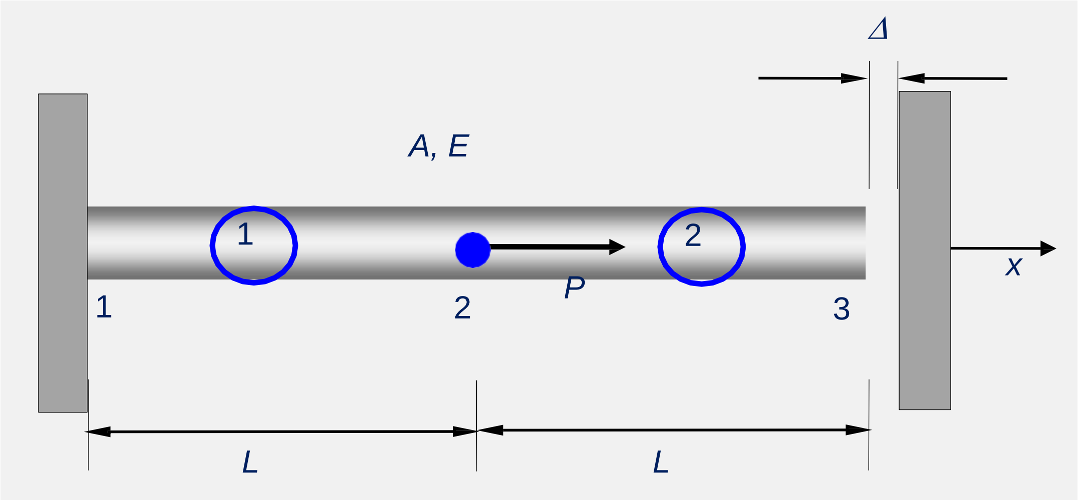
$$= E\frac{u_3 - u_2}{L} = \frac{E}{L}\left(0 - \frac{PL}{3EA}\right) = -\frac{P}{3A}$$

***Check the results*:** Draw the FBD and check the equilibrium of the structures.



Examples with Bar Elements

Example 2.2



Problem: Determine the support reaction forces at the two ends of the bar shown above, given the following

$$P = 6.0 \times 10^4 \text{ N}, \quad E = 2.0 \times 10^4 \text{ N/mm}^2,$$

$$A = 250 \text{ mm}^2, \quad L = 150 \text{ mm}, \quad \Delta = 1.2 \text{ mm}$$

Examples with Bar Elements

We first check to see if contact of the bar with the wall on the right will occur or not. To do this, we imagine the wall on the right is removed and calculate the displacement at the right end.

$$\Delta_0 = \frac{PL}{EA} = \frac{(6.0\times10^4)(150)}{(2.0\times10^4)(250)} = 1.8\text{mm} > \Delta = 1.2\text{mm}$$

Thus, contact occurs and the wall on the right should be accounted for in the analysis.

The global FE equation is found to be

$$\frac{EA}{L}\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix}$$

Examples with Bar Elements

The load and boundary conditions are

$$F_2 = P = 6.0 \times 10^4 \,\text{N}$$

$$u_1 = 0, \qquad u_3 = \Delta = 1.2\text{mm}$$

FE equation becomes

$$\frac{EA}{L}\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix} \begin{Bmatrix} 0 \\ u_2 \\ \Delta \end{Bmatrix} = \begin{Bmatrix} F_1 \\ P \\ F_3 \end{Bmatrix}$$

Solving this, we obtain

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 1.5 \\ 1.2 \end{Bmatrix} (\text{mm})$$

Examples with Bar Elements

To calculate the support reaction forces, we apply the 1st and 3rd equations in the global FE equation.

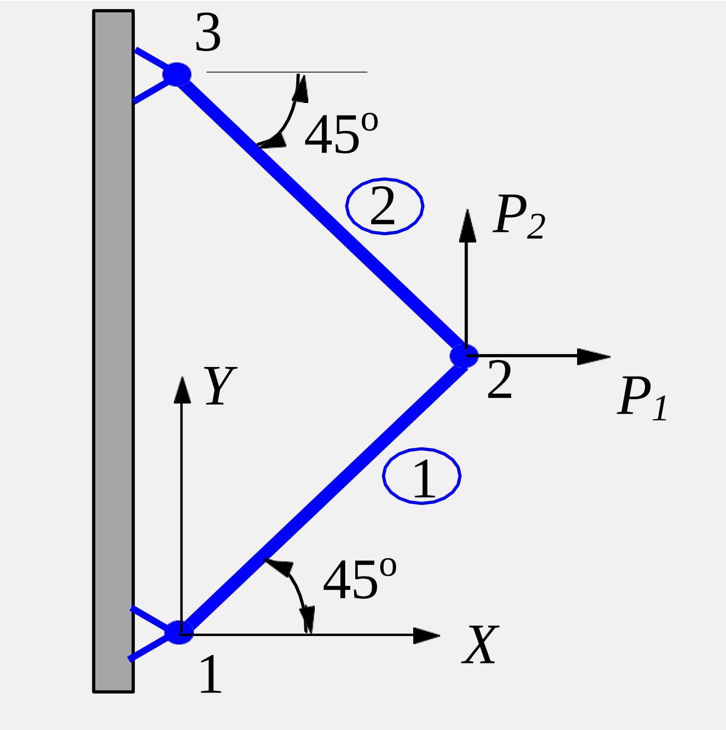
$$F_1 = \frac{EA}{L}\begin{bmatrix}1 & -1 & 0\end{bmatrix}\begin{Bmatrix}u_1\\u_2\\u_3\end{Bmatrix} = \frac{EA}{L}(-u_2) = -5.0\times10^4\,\mathrm{N}$$

$$F_3 = \frac{EA}{L}\begin{bmatrix}0 & -1 & 1\end{bmatrix}\begin{Bmatrix}u_1\\u_2\\u_3\end{Bmatrix} = \frac{EA}{L}(-u_2+u_3) = -1.0\times10^4\,\mathrm{N}$$

Check the results!

Examples with Bar Elements

Example 2.3



A simple plane truss is made of two identical bars (with *E, A,* and *L*), and loaded as shown in the above figure.

Find:

(a) displacement of node 2;
(b) stress in each bar.

Examples with Bar Elements

In local coordinate systems, we have

$$\mathbf{k}'_1 = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \mathbf{k}'_2$$

Element 1:

$$\theta = 45^o, \qquad l = m = \frac{\sqrt{2}}{2}$$

$$\mathbf{k}_1 = \mathbf{T}_1^T \mathbf{k}'_1 \mathbf{T}_1 = \frac{EA}{2L} \begin{array}{c} \begin{matrix} u_1 & v_1 & u_2 & v_2 \end{matrix} \\ \begin{bmatrix} 1 & 1 & -1 & -1 \\ 1 & 1 & -1 & -1 \\ -1 & -1 & 1 & 1 \\ -1 & -1 & 1 & 1 \end{bmatrix} \end{array}$$

Element 2:

$$\theta = 135^o, \qquad l = -\frac{\sqrt{2}}{2}, m = \frac{\sqrt{2}}{2}$$

$$\mathbf{k}_2 = \mathbf{T}_2^T \mathbf{k}'_2 \mathbf{T}_2 = \frac{EA}{2L} \begin{array}{c} \begin{matrix} u_2 & v_2 & u_3 & v_3 \end{matrix} \\ \begin{bmatrix} 1 & -1 & -1 & 1 \\ -1 & 1 & 1 & -1 \\ -1 & 1 & 1 & -1 \\ 1 & -1 & -1 & 1 \end{bmatrix} \end{array}$$

Examples with Bar Elements

Assemble the structure FE equation

$$
\begin{array}{cc}
 & \begin{array}{cccccc} u_1 & v_1 & u_2 & v_2 & u_3 & v_3 \end{array} \\
\dfrac{EA}{2L} & \begin{bmatrix} 1 & 1 & -1 & -1 & 0 & 0 \\ 1 & 1 & -1 & -1 & 0 & 0 \\ -1 & -1 & 2 & 0 & -1 & 1 \\ -1 & -1 & 0 & 2 & 1 & -1 \\ 0 & 0 & -1 & 1 & 1 & -1 \\ 0 & 0 & 1 & -1 & -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} = \begin{Bmatrix} F_{1X} \\ F_{1Y} \\ F_{2X} \\ F_{2Y} \\ F_{3X} \\ F_{3Y} \end{Bmatrix}
\end{array}
$$

Load and boundary conditions (BC)

$$u_1 = v_1 = u_3 = v_3 = 0, \quad F_{2X} = P_1, F_{2Y} = P_2$$

Condensed FE equation

$$\frac{EA}{2L}\begin{bmatrix} 2 & 0 \\ 0 & 2 \end{bmatrix}\begin{Bmatrix} u_2 \\ v_2 \end{Bmatrix} = \begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix}$$

Examples with Bar Elements

Solving this, we obtain

$$\begin{Bmatrix} u_2 \\ v_2 \end{Bmatrix} = \frac{L}{EA} \begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix}$$

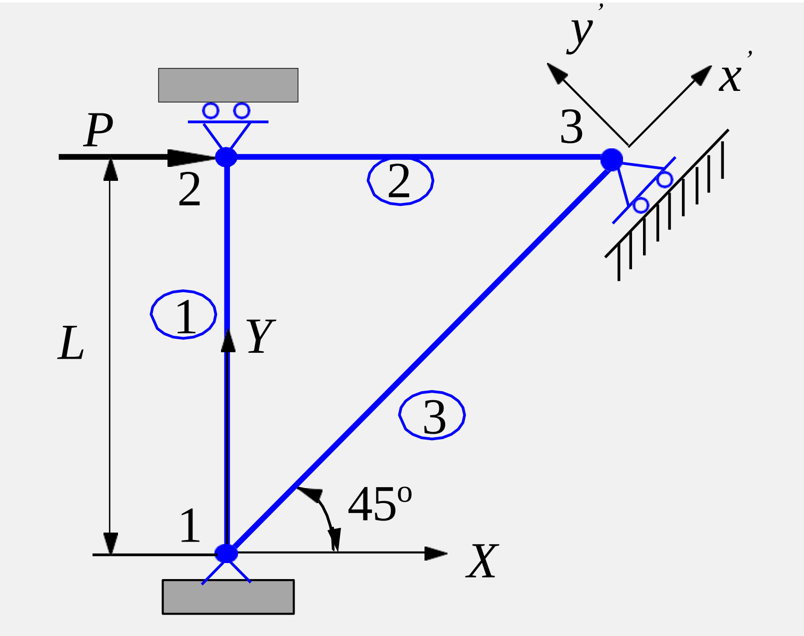
Stresses in the two bars

$$\sigma_1 = \frac{E}{L}\frac{\sqrt{2}}{2}\begin{bmatrix} -1 & -1 & 1 & 1 \end{bmatrix} \frac{L}{EA} \begin{Bmatrix} 0 \\ 0 \\ P_1 \\ P_2 \end{Bmatrix} = \frac{\sqrt{2}}{2A}(P_1 + P_2)$$

$$\sigma_2 = \frac{E}{L}\frac{\sqrt{2}}{2}\begin{bmatrix} 1 & -1 & -1 & 1 \end{bmatrix} \frac{L}{EA} \begin{Bmatrix} P_1 \\ P_2 \\ 0 \\ 0 \end{Bmatrix} = \frac{\sqrt{2}}{2A}(P_1 - P_2)$$

Examples with Bar Elements

Example 2.4 (Multipoint Constraint)



For the plane truss shown above,

$$P = 1000\ \text{kN}, \quad L = 1m,\ E = 210\,GPa,$$

$$A = 6.0 \times 10^{-4}\,m^2 \qquad \text{for elements 1 and 2,}$$

$$A = 6\sqrt{2} \times 10^{-4}\,m^2 \qquad \text{for element 3.}$$

Determine the displacements and reaction forces.

Examples with Bar Elements

Element 1: $\theta = 90^o, \qquad l = 0, m = 1$

$$\mathbf{k}_1 = \frac{(210\times10^9)(6.0\times10^{-4})}{1} \begin{array}{c} \begin{array}{cccc} u_1 & v_1 & u_2 & v_2 \end{array} \\ \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix} \end{array} \text{(N/m)}$$

Element 2: $\theta = 0^o,\ l = 1, m = 0$

$$\mathbf{k}_2 = \frac{(210\times10^9)(6.0\times10^{-4})}{1} \begin{array}{c} \begin{array}{cccc} u_2 & v_2 & u_3 & v_3 \end{array} \\ \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \end{array} \text{(N/m)}$$

Element 3: $\theta = 45^o, \qquad l = \frac{1}{\sqrt{2}}, m = \frac{1}{\sqrt{2}}$

$$\mathbf{k}_3 = \frac{(210\times10^9)(6\sqrt{2}\times10^{-4})}{\sqrt{2}} \begin{array}{c} \begin{array}{cccc} u_1 & v_1 & u_3 & v_3 \end{array} \\ \begin{bmatrix} 0.5 & 0.5 & -0.5 & -0.5 \\ 0.5 & 0.5 & -0.5 & -0.5 \\ -0.5 & -0.5 & 0.5 & 0.5 \\ -0.5 & -0.5 & 0.5 & 0.5 \end{bmatrix} \end{array} \text{(N/m)}$$

Examples with Bar Elements

The global FE equation is

$$1260\times10^{5}\begin{bmatrix} 0.5 & 0.5 & 0 & 0 & -0.5 & -0.5 \\ & 1.5 & 0 & -1 & -0.5 & -0.5 \\ & & 1 & 0 & -1 & 0 \\ & & & 1 & 0 & 0 \\ & & & & 1.5 & 0.5 \\ Sym. & & & & & 0.5 \end{bmatrix}\begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix}=\begin{Bmatrix} F_{1X} \\ F_{1Y} \\ F_{2X} \\ F_{2Y} \\ F_{3X} \\ F_{3Y} \end{Bmatrix}$$

Load and boundary conditions (BCs)

$$u_1 = v_1 = v_2 = 0, \text{and } v_3^{'} = 0,$$

$$F_{2X} = P, \qquad F_{3x'} = 0.$$

Examples with Bar Elements

From the transformation relation and the BC, we have

$$v'_3 = \begin{bmatrix} -\dfrac{\sqrt{2}}{2} & \dfrac{\sqrt{2}}{2} \end{bmatrix} \begin{Bmatrix} u_3 \\ v_3 \end{Bmatrix} = \frac{\sqrt{2}}{2}(-u_3 + v_3) = 0,$$

that is

$$u_3 - v_3 = 0$$

This is a *multipoint constraint* (MPC).

Similarly, we have a relation for the force at node 3

$$F_{3x'} = \begin{bmatrix} \dfrac{\sqrt{2}}{2} & \dfrac{\sqrt{2}}{2} \end{bmatrix} \begin{Bmatrix} F_{3X} \\ F_{3Y} \end{Bmatrix} = \frac{\sqrt{2}}{2}(F_{3X} + F_{3Y}) = 0,$$

that is

$$F_{3X} + F_{3Y} = 0$$

Examples with Bar Elements

Applying the load and BC's in the structure FE equation by "deleting" the 1st, 2nd and 4th rows and columns, we have

$$1260\times10^5\begin{bmatrix}1 & -1 & 0\\ -1 & 1.5 & 0.5\\ 0 & 0.5 & 0.5\end{bmatrix}\begin{Bmatrix}u_2\\ u_3\\ v_3\end{Bmatrix}=\begin{Bmatrix}P\\ F_{3X}\\ F_{3Y}\end{Bmatrix}$$

Further, from the MPC and the force relation at node 3, the equation becomes

$$1260\times10^5\begin{bmatrix}1 & -1 & 0\\ -1 & 1.5 & 0.5\\ 0 & 0.5 & 0.5\end{bmatrix}\begin{Bmatrix}u_2\\ u_3\\ u_3\end{Bmatrix}=\begin{Bmatrix}P\\ F_{3X}\\ -F_{3X}\end{Bmatrix}$$

which is

$$1260\times10^5\begin{bmatrix}1 & -1\\ -1 & 2\\ 0 & 1\end{bmatrix}\begin{Bmatrix}u_2\\ u_3\end{Bmatrix}=\begin{Bmatrix}P\\ F_{3X}\\ -F_{3X}\end{Bmatrix}$$

Examples with Bar Elements

Solving this, we obtain the displacements

$$\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \frac{1}{2520\times 10^5}\begin{Bmatrix} 3P \\ P \end{Bmatrix} = \begin{Bmatrix} 0.01191 \\ 0.003968 \end{Bmatrix} (\mathrm{m})$$

From the global FE equation, we can calculate the reaction forces

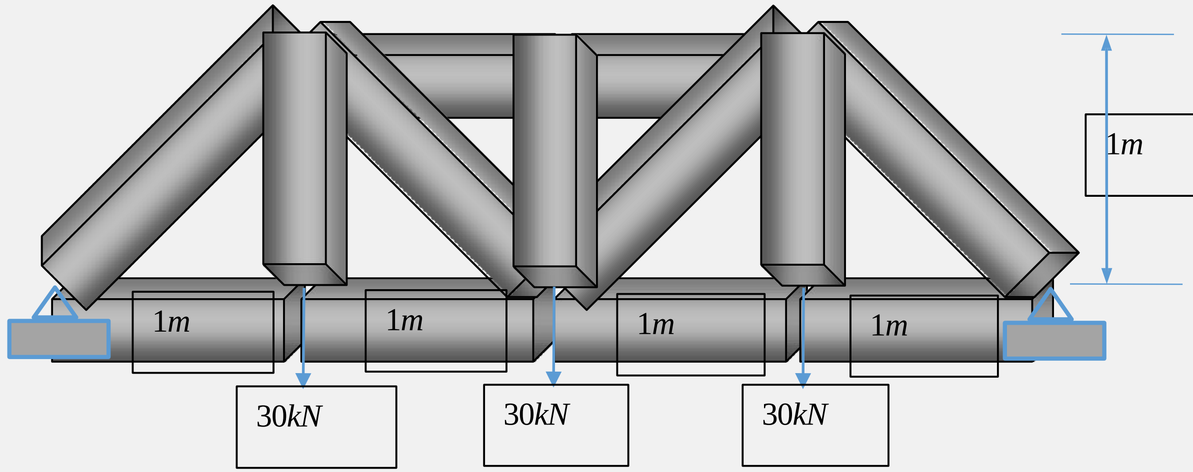
$$\begin{Bmatrix} F_{1X} \\ F_{1Y} \\ F_{2Y} \\ F_{3X} \\ F_{3Y} \end{Bmatrix} = 1260\times 10^5 \begin{bmatrix} 0 & -0.5 & -0.5 \\ 0 & -0.5 & -0.5 \\ 0 & 0 & 0 \\ -1 & 1.5 & 0.5 \\ 0 & 0.5 & 0.5 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \\ v_3 \end{Bmatrix} = \begin{Bmatrix} -500 \\ -500 \\ 0.0 \\ -500 \\ 500 \end{Bmatrix} (\mathrm{kN})$$

Analysis of a truss bridge …

Case Study with ANSYS Workbench

Case Study with ANSYS Workbench

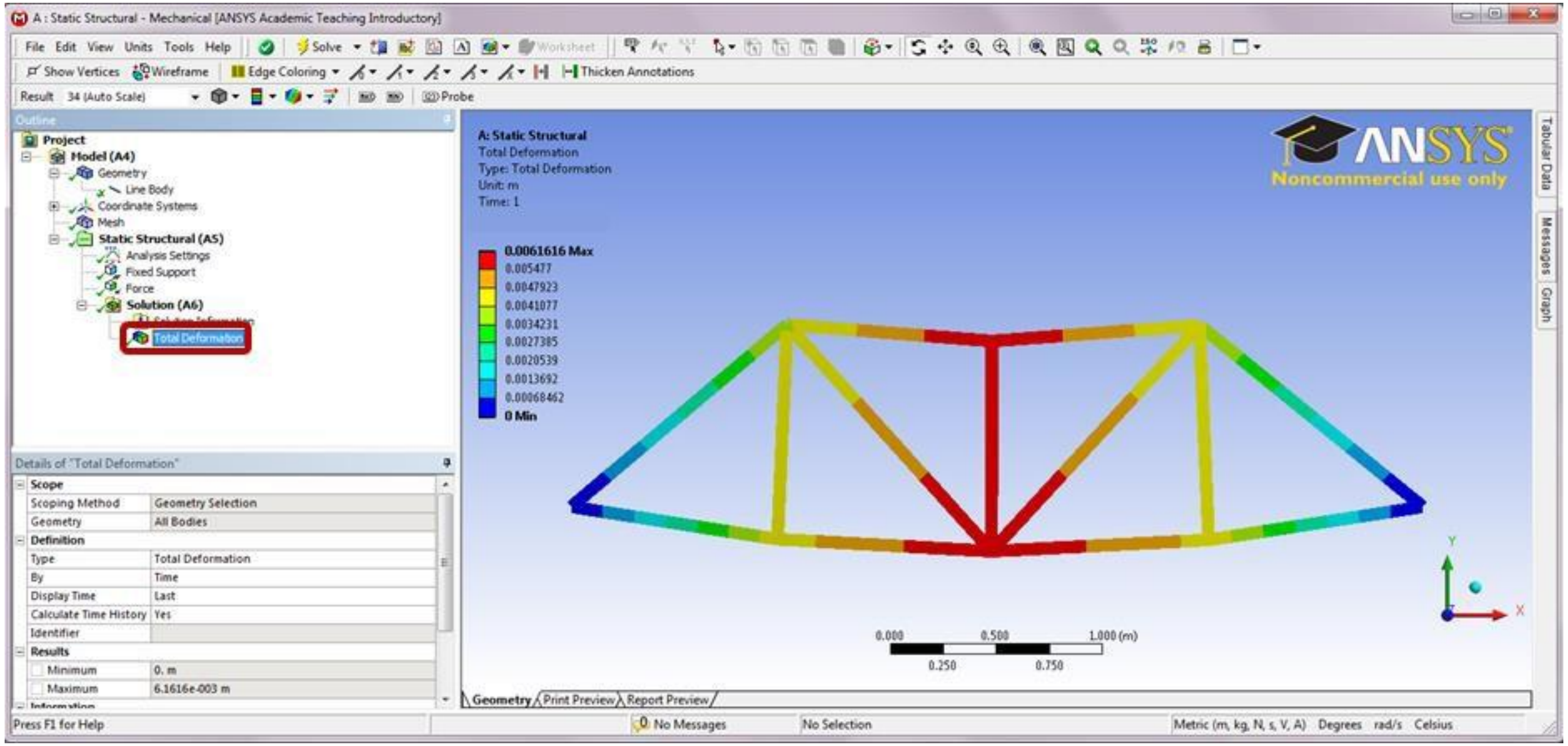
<Problem Description> Truss bridges can span long distances and support heavy weights without intermediate supports. They are economical to construct and are available in a wide variety of styles. Consider the following planar truss, constructed of wooden timbers, which can be used in parallel to form bridges. Determine the deflections at each joint of the truss under the given loading conditions.



Material: Douglas Fir
$E = 13.1\ GPa$
$\nu = 0.29$
Member cross section:
height = 6 cm
width = 6 cm

Case Study with ANSYS Workbench

Run a ***Static Structural Analysis*** to review the truss deformation results.



Summary

In this chapter,

- We study the bar elements which can be used in truss analysis.
- The concept of the shape functions is introduced and the derivations of the stiffness matrices using the energy approach are introduced.
- Treatment of distributed loads is discussed, and several examples are studied.
- A planar truss structure is analyzed using *ANSYS Workbench*. It provides basic modeling techniques and shows step-by-step how *Workbench* can be used to determine the deformation and reaction forces in trusses.